www.rericjournal.ait.ac.th

# The Influence of Moving Deflector Angle to Positive Torque on the Hydrokinetic Cross Flow Savonius Vertical Axis Turbine

Bagus Wahyudi *[1] and Satworo Adiwidodo*

***Abstract –*** *Nowadays, the experts are being chased by time to find a substitution for fossil energy and develop the use of renewable energy that is environmentally friendly. This study introduces the new model of Vertical Axis Hydrokinetic Turbine using a pair of a savonius tandem blade (STB). When the flow is coming onto the left side of the vertical axis turbine, the rotor will be rotated clockwise, while the flow on the right side of the rotor will become an obstacle of its rotation due to a negative torque generated by the flow itself. The aim of developing a new model of this turbine is to counter the negative torque which is usually common in the vertical axis turbine and also to improve the efficiency. The research methodology is using CFD Simulation and the Response Surface Method (RSM). Variation of the angle of moving plate deflector (MPD) is (a) the radial MPD, and (b) the tangential MPD. Based on simulation and analytical review proved that a pair of STB which is combined with the tangential MPD has a higher capability to increase the positive torque than that combined with the radial MPD.*

***Keywords –*** CFD simulation, hydrokinetic turbine, moving deflector, RSM optimization, tandem Savonius.

## 1. INTRODUCTION

Vertical axis turbines are less efficient compared to horizontal axis turbines. However, vertical axis turbines can deliver fluid flow from all directions. Vertical axis turbines are usually used for small-scale power plants and for power generation applications in remote locations. Savonius turbine, Gorlov turbine (the turbine helix), Darrieus turbine, and turbine Darrieus type-H are commonly used as vertical axis kinetic turbine. Low efficiency of vertical axis turbine is commonly caused by the flow that is split into two sides on the left and right side of the shaft. When the flow is coming onto the left side of the rotor, the rotor will be rotated clockwise, while the flow on the right side of the rotor will become an obstacle of its rotation due to a negative torque generated by the flow itself. Therefore, the earlier researchers initiated a way to overcome the negative torque on the vertical axis water turbine using deflector plate [1] and also Guide Box Tunnel [2].

Despite the disadvantages, Savonius rotor has many benefits compared to the others because of its simple and low-cost construction, its ability to adjust the flow direction, and its characteristic of having a good starting torque at lower speeds. For those reasons, many studies have been conducted by researchers to increase the performance of Savonius turbine rotor. This project develops a new model of hydrokinetics savonius tandem blade of cross flow vertical axis turbine in the hope of improving their performance and efficiency.

The application of computational fluid dynamics (CFD) was grown quickly in the last decade of numerical flow simulation on the design of hydraulic machinery. Now, CFD simulations can even manage complex geometries from entire machines with high-precision [3]. By using CFD, the present research is focused on modification design of a rotor blade shape using a numerical optimization method coupled with the statistical approach. Response surface method (RSM) is a combination of statistical and mathematical techniques useful for increasing the optimization process, which uses regression analysis, analysis of variance, and design of experiment (DOE).

Several studies for increasing turbine performance of Savonius with an original rotor blade (a pair of semi-circular blades) have been reported by a number of researchers, such as Fujisawa [4], Nakajima [5], and others. Deda Altan has already tested the Savonius turbine using curtain (like deflector) on wind tunnel resulting power coefficient of 38.5% for the semi-circular blade [6], and Khan also reporting several tests on variant blade shapes with 35% power coefficient result [7].

However the effect of geometrically and interactions between aerodynamic parameters on the rotor was not observed in detail in these performance studies. The experimental results of various savonius wind turbines which modified have not shown a significant increase due to fluid weight effect. From this point of view, so many researchers have explained the influence of water as working fluid on the Savonius turbine [8]-[14].

The original Savonius rotor as shown in Figure 1 has two pairs of cylindrical blades that look like a letter S which are not connected in the middle (overlapping in the middle) on both ends of the blade so that serve as the entry of outflow from the first blade (advancing blade) to the second blade (returning blade). As shown in Figure 4, the Savonius tandem blade (STB) overlapping with the first blade and also on the second blade generating a returned force from the opposite direction outflow through overlap compartment can increase the aggregates torque.

*Department of Mechanical Engineering, State Polytechnic of Malang, Campus : Jl. Soekarno-Hatta No. 9, Malang 65141, Indonesia..

[1]Corresponding author:
Tel.: +62 817 535 222; Fax: +62 341 404 420
E-mail: bagus.wahyudi@polinema.ac.id

The STB was made to optimize the design of the original Savonius rotor in order to improve the efficiency. The efficiency of Savonius rotor depends on the pressure difference between the convex and concave part of the blade. The pressure on concave side must be higher than convex side in order to generate positive torque.

The previous study has results the best design of diffuser Savonius Tandem Blade (STB) using CFD simulation. The result of study producing three models of tandem blade *i.e.*: (a) Tandem Overlap TO, (b) Tandem Symmetrically TS and (c) Tandem Convergent TC, one of the best designed to be deeply studied has already obtained. The benchmarking of the three model above, the convergent model (TC) shows very suitable to generate hydropower by evaluating the maximum pressure gap (Δp) between upstream and downstream [15].

Golecha has already improved the performance of the S-shaped rotor savonius by adding plates to deflect or redirect the flow of which is placed in front of the rotor. By using an adjustable tip angle of deflector until reaching the optimal position, it is proved that the power coefficient (Cp) is increased by about 50% (see Figure 2).

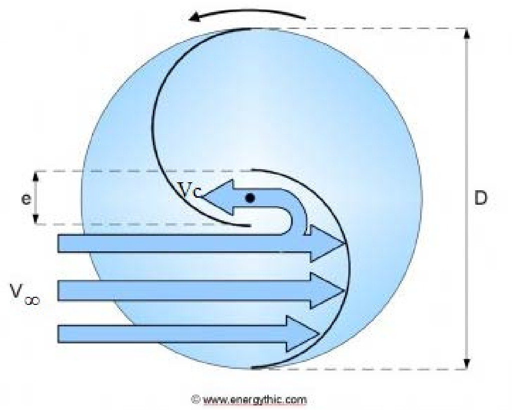


**Fig. 1. The original savonius rotor with overlapping in the middle.**

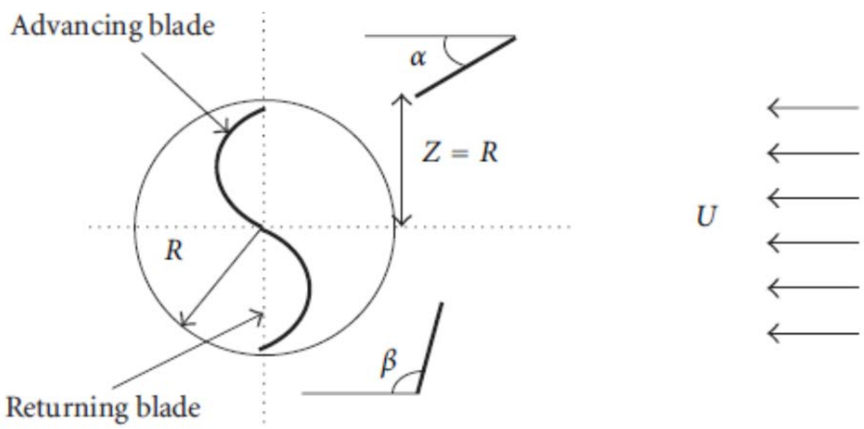


**Fig. 2. The configuration of guide flow on savonius rotor using deflector plate by Golecha [2].**

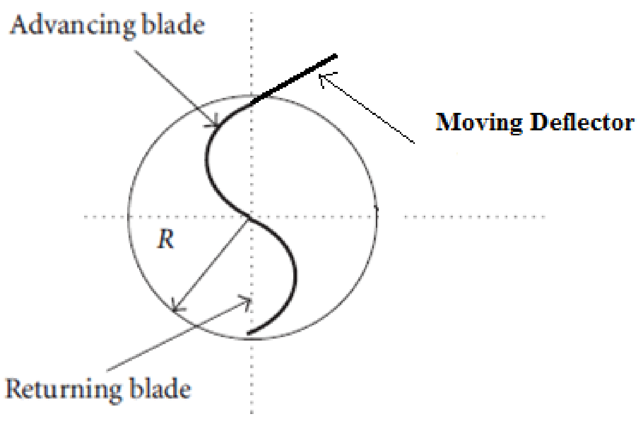


**Fig. 3. Combine savonius rotor with moving deflector.**

This study promotes the new parts in design to improve the performance of Hydrokinetics Cross Flow Vertical Axis Turbine which consists of Savonius Tandem Blade (STB) and Moving Plate Deflector (MPD). This new design has a high solidity and broader swept area compared to savonius conventional rotor due to the increase of torque on turbine rotor.

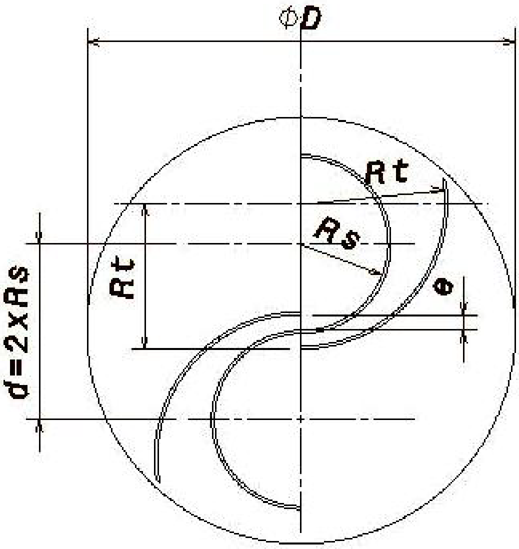


**Fig. 4. STB (a pair of nozzle and diffuser) set up in the central part of the rotor has to function to create the cross flow and to increase torque [16].**

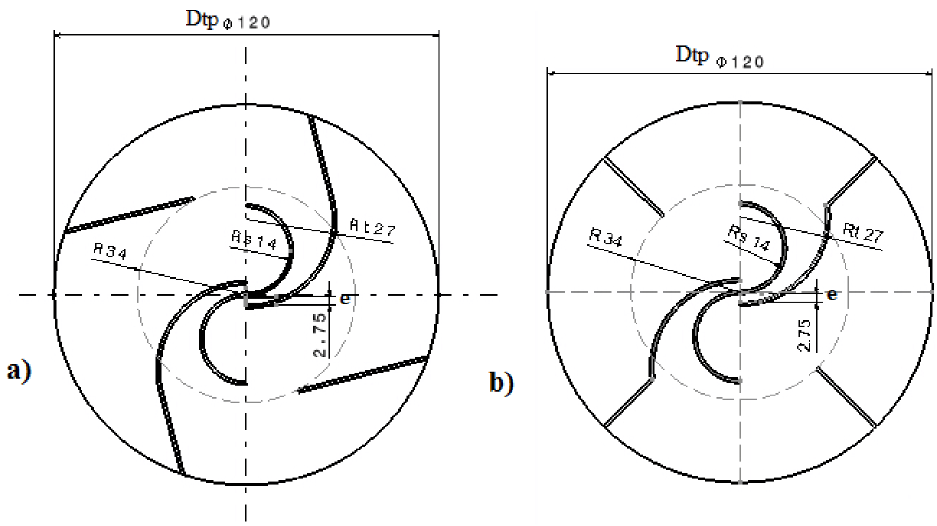


**Fig. 5. Hydrokinetics Cross Flow Vertical Axis Turbine is a combination of STB in the centre of rotor turbine with (a) Tangential MPD or (b) Radial MPD in the outside of tip diameter.**

## 2. THE CONCEPT OF CROSS FLOW SAVONIUS VERTICAL AXIS TURBINE (CROSSVAT)

This study introduces the new model of Vertical Axis Hydrokinetic Turbine using a pair of the STB. The STB which is built like a pair of nozzle and diffuser in the central part of the rotor has a function to change the direction of flow (returning flow to backward) and also to reinforce the pressure on the surrounding blades. The aim of developing a new model of this turbine is to counter the negative torque which is usually common in the vertical axis turbine and also to improve the efficiencies.

The first step on the Savonius rotor design is broadening swept area with installing tandem blade near from S-blade that can increase drag force production on the blade surface as shown in Figure 4. To achieve the maximum power generation based on preliminary design study we must use the convergent blade design [15]. The CFD simulation and RSM are used to realize optimum Radius Tandem Blade (Rt) and overlap width (e) as the independent variable. A tandem blade convergent model has been selected for a preliminary design study for the three models based on pressure gap maximum between upstream and downstream by using CFD simulations. This study used the nozzle-diffuser cross flow intermediary model as seen in Figure 4 as the basic form of a STB that will be developed and optimized further by adding the plate deflector. This model is expected to have a maximum Rcv and able to increase the torque on the rotor [16].

The second step is assembling the deflector on the end of a tandem blade tip radial direction along the 1/2 (Dtp-D) as seen as Figure 5. The distance between the tip of the axis of the deflector plate is Tip Radius (Rtp) and the plate deflector angle (β) is set as the dependent variable, while the response variable is the gap pressure (Δp) between the downstream and upstream.

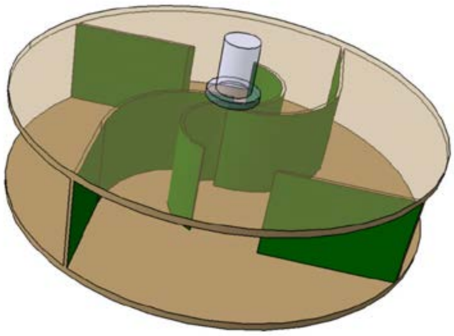

**Fig. 6. Rotor Turbine Design Using Combination STB and MPD Tangential.**

To find the torque and drag characteristics in the normal and tangential direction of the chord, the values of $F_n$ and $F_t$ must be known before. The values of the forces $F_n$ and $F_t$ are obtained by integrating the pressure for a blade.

$$F_n = \int_0^n \Delta P.H.\frac{d}{2}\cos\phi \, d\phi = \sum_{i=1}^n \Delta P_i.H.\frac{d}{2}\cos\phi_i \Delta\phi_i \tag{1}$$

$$F_n = \sum_{i=1}^n \Delta P_i H \frac{d}{2}\sin\phi_i \Delta\phi_i \tag{2}$$

Where $\Delta p_i$ is the difference between the pressure on the concave and convex surfaces at a particular pressure tapping, $F_n$ plays a role in producing a torque on the shaft of the rotor and this torque can be expressed as Torque on a blade, [17]-[18].

$$T = \sum_{i=1}^n R \times F_n = \sum_{i=1}^n R \times \Delta P_i H \frac{d}{2}\sin\phi_i \Delta\phi_i$$

$$T = \sum_{i=1}^n \left[ R_i \times \int_{\phi_i}^{\phi} \Delta P_i H \frac{d}{2}\sin\phi_i \Delta\phi_i \right] \tag{3}$$

where: Sd = width of blade ; H = height ; $\Delta p_i$ = gap pressure between concave and convex of blade surface ; and $R_i$ = distance from centre of .gravity on each blade surfaces.

For a fluid flowing through the cross section A, the mass flow rate is ρ.A.V, and therefore the power is:

$$\Delta P = \rho.AV.1/2..V^2 = 1/2.\rho.AV^3 \tag{4}$$

Where Ps is the power shaft (watts), ρ is the fluid density (kg/m3), and V is the working fluid speed (m/s). The power is the energy flux or power density of fluid flow through the cross section (surface area). The ratio of shaft power (Ps) to the power available in the fluids flow (Pf) is known as the coefficient of power (Cp = Ps/Pf) and this indicates the efficiency of energy conversion.

By using Phi Buckingham method, a configuration of dimensionless parameter groups that influence efficiency or coefficient of power (Cp) is obtained as shown in Equation 5.

$$Cp = \xi\left[\frac{H}{D}, \frac{Rt}{D}, \frac{e}{D}, \frac{Sd}{D}, \frac{Vc}{U}, \lambda, \frac{\Delta p\infty}{\rho U^2}\right] \tag{5}$$

Where:

| | | |
|---|---|---|
| H/D | = | Aspect Ratio |
| Rt/D | = | Tandem Ratio |
| e/D | = | Overlap Ratio |
| Sd/D | = | Moving deflector angle |
| Vc/U | = | Clearance velocity ratio (Rcv) |
| λ | = | Tip Speed Ratio |

## 3. METHOD

The effect of the fast growing application of computational fluid dynamics (CFD) within the last two decades is the significance of numerical flow simulations in the design of hydraulic machinery that has grown to a considerable extent. At present, CFD simulations can often replace laboratory experiments due to the fact that even complex geometries and entire machines can be modelled. Many types of research were conducted to demonstrate the influence of modified Savonius rotor blade geometry parameters such as twist blade, overlap ratio, a number of blades, multi-stage, sweep area, non-circular blade, and additional guide blade on the aerodynamic performance of the rotor blade. However, the influence of geometrical design variables and their interactions on the rotor aerodynamic performance was not examined in detail in these works. From this point of view, the present research is focused on suggesting a rotor blade shape design using the numerical optimization method coupled with the statistical approach.

A two-dimensional view of the rotor model was considered. It is because the rotor blades rotate in the same plane as the approaching water flow stream. The computational domain was discretized using two-dimensional unstructured mesh (triangular mesh). The left boundary had Velocity Inlet condition while the right boundary had Outflow condition. The top and bottom boundaries for the open channel sidewalls had symmetrical conditions. The moving wall condition was employed for the rotor model to study the effect of fluid motion in and around the rotating Savonius rotor. The dimensions of the computational domain were 500 mm in length and 75.5 [mm] in widths, which were also similar to the experimental conditions. For the various model conditions, the geometry of the rotor was changed and accordingly different meshes were generated for each condition.

Steps in the CFD simulation solutions consist of:

1. Mesh Report: Nodes 4956, Element 3374

2. Solver (Pressure Based, Steady, 2D)
3. Viscous Model: Standard k-epsilon (k-ε) / Near-Wall Treatment: Standard Wall Functions
4. Material: Water (μ= 1.002x10-3 [kg/m-s], ρ= 1.000 [kg/m3])
5. Operating conditions:
   Atm. Pressure (1.0132 [bar])
   Temperature 25$^o$C
6. Boundary Conditions:
   - Inlet: Velocity Inlet
   - Sides: No slip wall
   - Blades: Stationary Wall
   - Outlet: Outlet
7. Solution controls:
   - Pressure-Velocity Coupling: SIMPLE
   - Discretization: fluids
8. Pressure (Standard) / Inlet Velocity: 1 [m/s]

The optimization process using Response surface method (RSM) for designing geometry of Convergent STB is shown in Figure 4 above. RSM is a collection of statistical and mathematical techniques useful for developing and improving the optimization process, which collectively uses the design of the experiment, regression analysis, and analysis of variance. In this case, the response variable pressure gap (y) is affected by two independent variables: Radius Tandem Blade "Rt" or (x1) and Clearance Blade "e" or (x2). By using an appropriate model formulation, the optimal value of independent variables (x1, x2) which causes maximum pressure gap can be obtained.

The subsequent RSM analysis can be used for surface fitting. If the surface fitting is a good approximation of a function of the response, the surface fitting analysis would be equivalent to the actual analysis systems. Analysis of variance and regression analysis are the statistical techniques to estimate regression coefficients in the quadratic polynomial model and also yield a measure of uncertainty in the coefficients.

Furthermore, at the condition near the response, the second order model (order II) or more is normally required to approximate the response due to the curvature of the surface. In many cases, the model order II two expressed in Equation 6 is considered sufficient.

$$\hat{y} = \beta_0 + \sum_{i=1}^{k} \beta_i x_i + \sum_{i=1}^{k} \beta_{ii} x_i^2 + \sum_i \sum_j \beta_{ij} x_i x_j, i < j \tag{6}$$

Analysis of the response surface fitting Order II is often referred as the canonical analysis. The least squares method is used to estimate parameters on the approximation functions. One of the important statistical parameters is the coefficient of determination, $R^2$ which provides the summary statistic that measures how well the regression equation fits the data.

However, a large value of $R^2$ does not necessarily imply that the regression model is a good one. Adding a variable to the model always increases $R^2$, regardless of whether the additional variable is statistically significant or not. Thus, it is possible for the models that have large values of $R^2$ to yield the poor predictions of new observations or estimates of the mean response.

## 4. RESULTS AND DISCUSSIONS

By using simulation software ANSYS Release 14.5 it's the following boundary conditions that have been applied. The stationary domain has a free stream velocity. The hydrodynamic pressure conditions are applied and the initialization is done. Inlet and Outlet are default boundary conditions in simulation software. Inlet requires the speed of inlet velocity of water and the outlet requires the relative pressure, 1.0132 x 105 [Pa], at the initial conditions. The blade surfaces can be used as a "wall" condition. This condition enables the calculation of properties such as force on the surface. Once the domains have been specified, a default fluid-fluid interface is detected between the rotating and stationary domain.

The dimensions of the computational domain were 500 mm in length and 75.5 [mm] in width for simulation of basic STB (Figure 4) and 150 [mm] in width for simulation of turbine rotor with moving plate deflector (Figure 5), which were also similar to the experimental conditions. For the various model conditions, the geometry of the rotor was changed and accordingly different meshes were generated for each condition.

The contour plots as shown in Figures 9 to 12 predict the variations in velocity and pressure in various regions near the blades within the flow domain. It can be observed from the pressure contour plots that pressure gap occur across the rotor from upstream to downstream side. This pressure gap indicates power extracted by the rotor causing its rotation.

The static pressure on the convex side of the blades can be observed to be lower than those on the concave side of the blades; in fact, a region of negative pressure exists on the convex side of the blades. This occurs due to the high flow velocity over the convex side of the blades. As a result, a pressure gap acts across the concave and convex side of the blades, which provide the necessary torque for causing rotation as seen as in Equation 3 [17], [18].

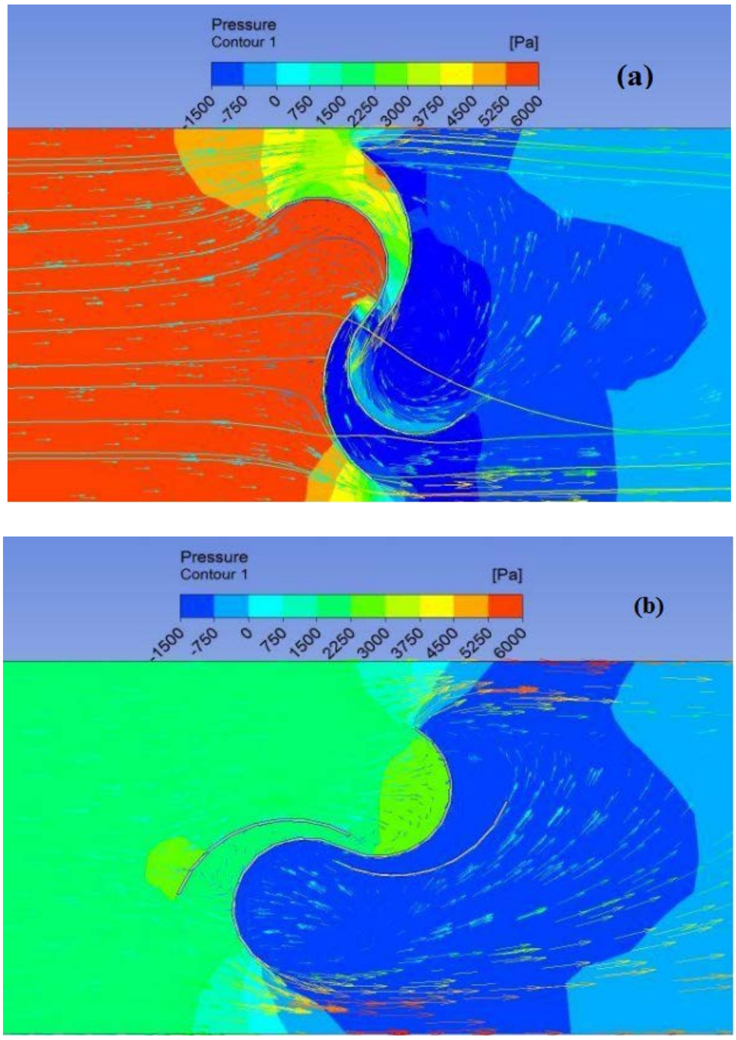


**Fig. 7. Pressure contour and streamline of STB rotor model on rotate position of 135$^{o}$ (a) and 210$^{o}$ (b).**

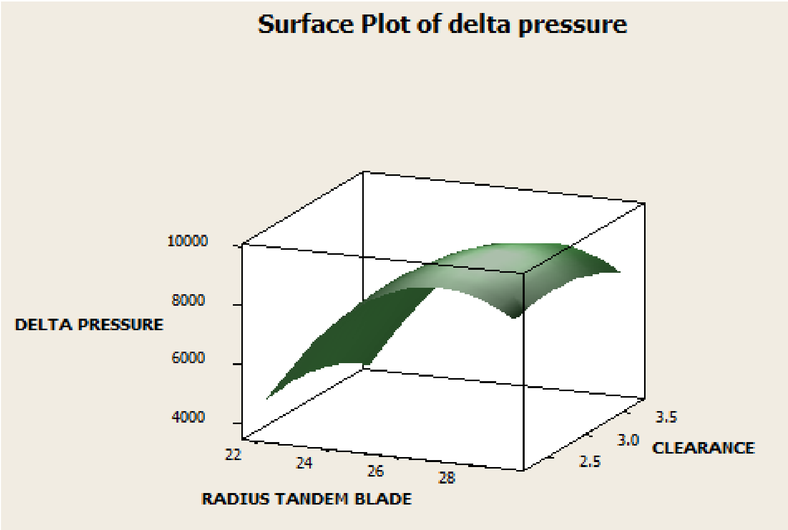


**Fig. 8 . RSM Optimum result of STB model.**

From the contour plot of the image above, it can be concluded that the image does not have a stationary point. Consequently, calculation of the stationary point and surface characteristics of the response is not necessary.

The result of response (ΔP) optimization will be obtained after value of (Rt) and (e) are substituted into the model Equation (7) with constant value of Rt = 27 [mm] and variable e = 2.5 [mm] ÷ 3.5 [mm]. By using Minitab Software, it is simpler to obtain the result of peak point in the tangential as shown in Figure 8 which shows an optimal value of Rt = 27 [mm] and e = 2.75

[mm] with the response of max. $\Delta P = 9415.91$ [Pa].

$$\Delta P = 133732 + 10071\,Rt - 5318.11\,c + 186.77\,Rt^2 + 965.99\,c^2 \tag{7}$$

The next step is to develop STB geometry by adding a deflector plate on the outer edge. Provided two design models namely deflector plate with the radial direction (model 2) and deflector plate with the tangential direction (model 1). Size diameter of rotor turbine is to set a maximum of 140 mm and a minimum of 100 mm diameter tip. The moving deflector is set on the maximum angle of 46$^{o}$ (tangential direction or Sd/D=0,295) and a minimum of 0° (radial direction or Sd/D=0,257). Some simulated data with tip diameter and angle of plate deflector are shown in Figures 9 to 12.

At this point of view, it is very important to investigate the ratio of the clearance velocity (Rcv = Vc / V∞) in order to know the contribution of kinetic energy change to strike the returning blade. Clearance "e" will give "jet effect" and increase the momentum force on the concave surface, which consequently will increase the torque on returning blade.

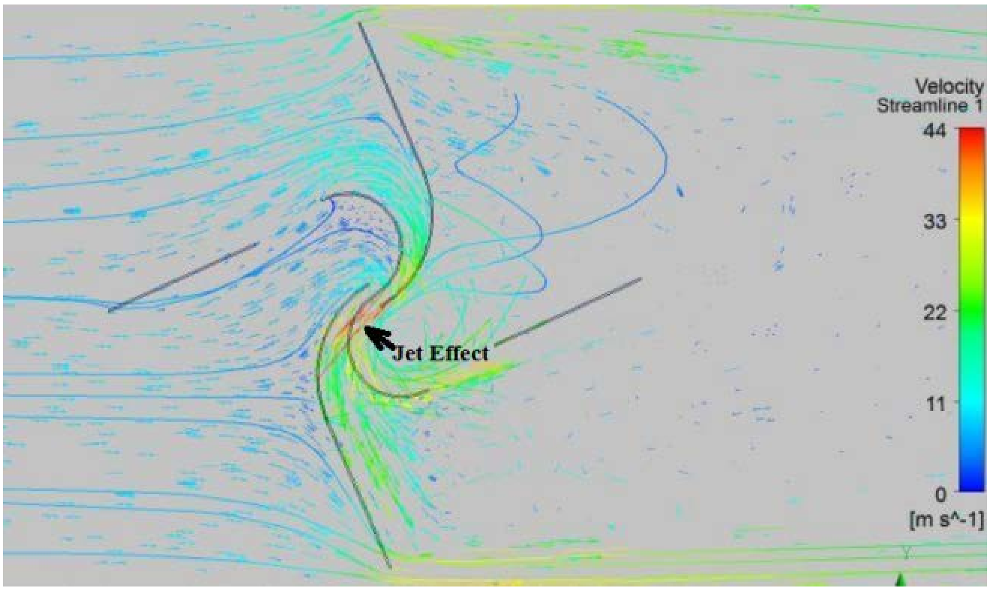


**Fig. 9. CFD simulation of streamline of CROSSVAT rotor model with tangential plate deflector (Sd/d= 0,295).**

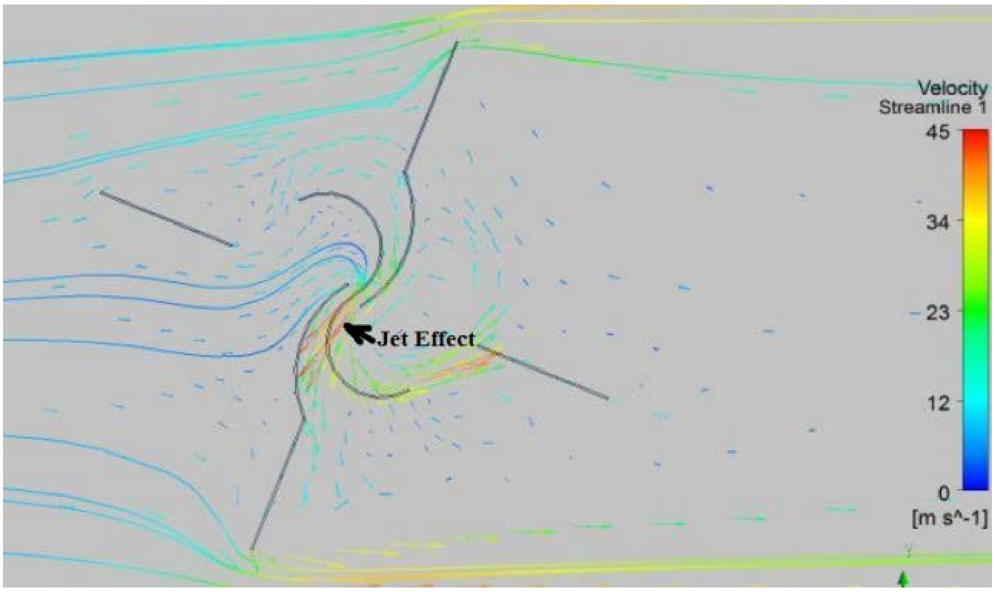


**Fig. 10. CFD simulation of streamline of CROSSVAT rotor model with radial plate deflector (Sd/D= 0,257).**

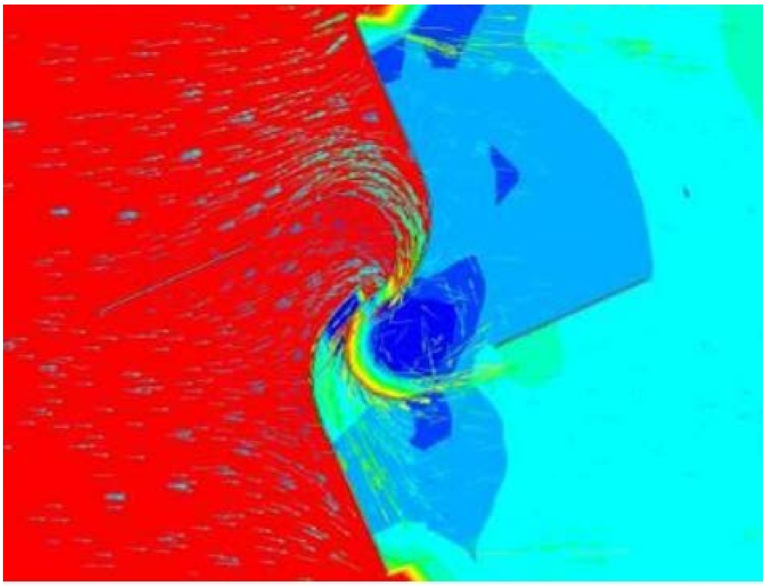

**Fig. 11. Pressure contour of CROSSVAT rotor model with tangential plate deflector (Sd/D= 0,295).**

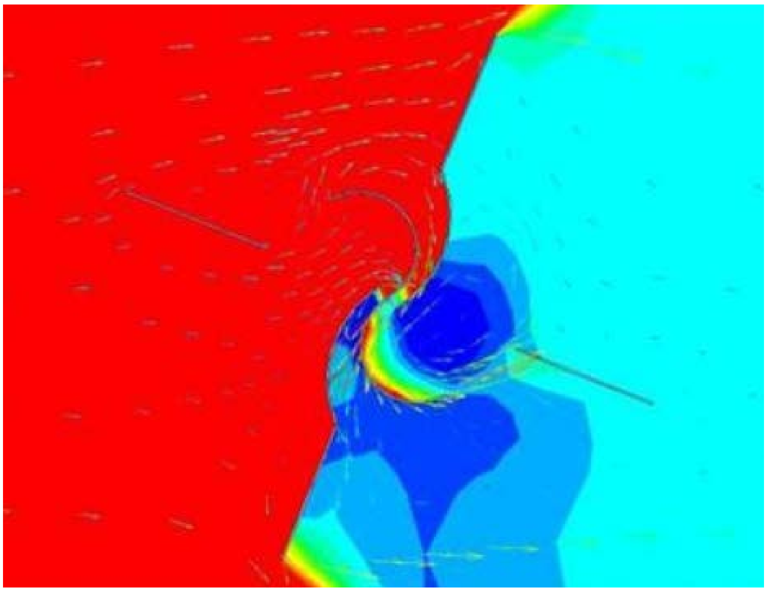

**Fig. 12. Pressure contour of CROSSVAT rotor model with radial plate deflector (Sd/D= 0,257).**

To understand the influence of the angle of deflector on the torque, Figure 13 explains that the tangential moving plate deflector (MPD) is more effective in generating positive torque than the radial deflector. By using CFD simulation, it is shown that the negative torque is commonly caused by blocking flow that occurred when the edge of the radial deflector is near from the wall position. At the rotated position of $150^{o}$ the two models result from the contra productive value; the tangential model resulted in the maximum positive torque but the radial model is the opposite.

The simulation of the tangential type of CROSSVAT Turbine model is shown in Figure 9, when the deflector plate is on the inclined position which makes it very effective for fluid flowing through the narrow compartment that plays a role as a nozzle on the advancing blade and serves as a diffuser on the returning blade. The jet effect create through the STB as seen on simulation study (Figure 9) is the reason for that increase in the positive torque as shown in Figure 13. At this point of view, the tangential model is more powerful in generating torque than the radial model. Torque due to the coupling force between the advancing blade and returning blade will be more balanced than the original savonius.

In laboratory experiments, a pair of dynamometer is used to measure forces of tension on the rope rubbing against the rotor pulley (Figure 14). A pair of coupling force measured on the dynamometer can be used to calculate the torque produced by the turbine shaft. Based on experimental test being done in laboratory, there are results graph of the Efficiency versus Tip Speed Ratio (TSR) as shown on Figure 15 that the turbine rotor using a combination of STB + MPD tangential to have higher efficiency than rotor using combination of STB + MPD radial.

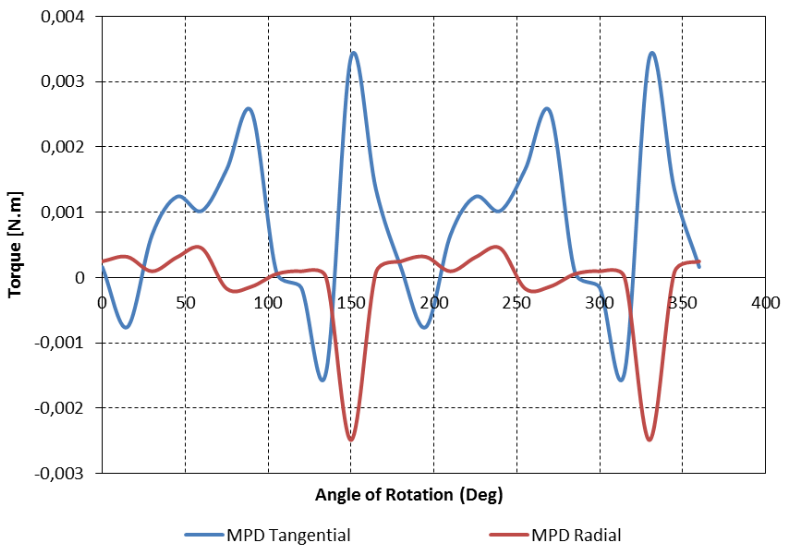


**Fig. 13: Torque Distribution vs. Angle of Rotation.**

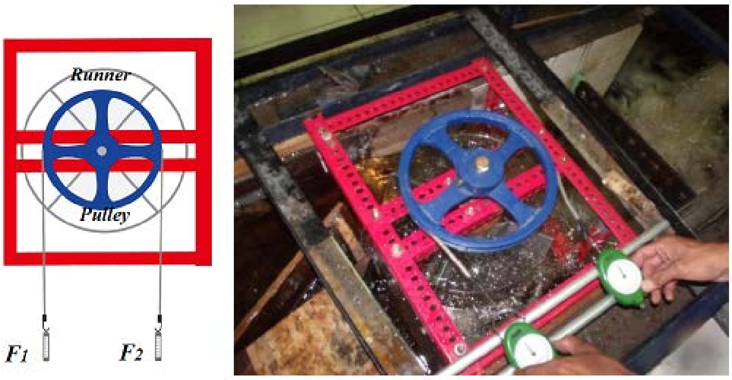


**Fig. 14: Measuring the Torque with dynamometer.**

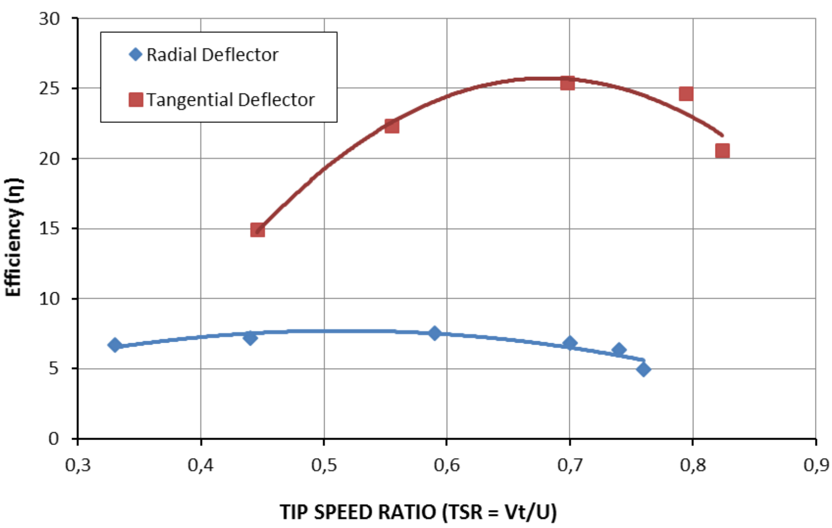


**Fig. 15: The performance result of CROSSVAT with Radial MPD and Tangential MPD by experimental test.**

In fact to achieve Savonius hydrokinetic turbine efficiency above 35% is difficult, but compared to the wind turbine, the type of hydrokinetic more smooth, a good service life and stable operation [19]. Compare with Akwa's research on Savonius wind turbine with the overlap ratio of 0:15 only produces about 31% efficiency [20], the CROSSVAT hydrokinetic turbine discussed in this paper is still more economically competitive. The idea of this type of turbine design is to be applied to the potential of the river with low head elevation which have been less be concerned. In techno-economic analysis, the application of hydrokinetic turbine stand-alone on the river was less advantageous economically, but if built some of the turbine generator set with synchronization or hybrid system would be cheaper and profitable [21], [22].

## 5. CONCLUSION

A. The design of the Savonius Tandem Blade combined with tangential moving plate deflector (STB + MPD tangential) when tested by CFD simulation testing showed good performance mainly to generate torque positive is greater than the Savonius tandem blade combined with radial moving plate deflector (STB + MPD radial).

B. The CROSSVAT hydrokinetic when tested through experimental testing proved that the STB + MPD tangential to have higher efficiency than STB + MPD radial.

C. Both based on CFD simulation and experimental testing in the laboratory, proving that there are have significant influence of moving deflector angle to the turbine performance. Thus the results of this study have been validated and can be used as a reference for subsequent research.

## ACKNOWLEDGEMENT

The authors are grateful to Directorate of Research and Community Service (DRPM) of Indonesian Ministry of Technological Research and High Education for financial support on the activities Research Grant in State Polytechnic of Malang during March until November 2016.

## REFERENCES

[1] Golecha K., Eldho T.I. and Prabhu S.V., 2011. Influence of the deflector plate on the performance of modified Savonius water turbine. *Applied Energy* (88): 3207-3217.

[2] Golecha K., Eldho T.I. and Prabhu S.V., 2012. Study on the interaction between two hydrokinetic Savonius turbines. *International Journal of Rotating Machinery*, Vol. 2012, Article ID 581658, 10 pages.

[3] Kumar M.S and P. Philominathan. 2011. Bringing out fluids experiments from laboratory to in Silico. A journey of hundred years. *American Journal of Computational Mathematics* (1): 271-280.

[4] Fujisawa N. and F. Gotoh. 1994. Experimental study on the aerodynamic performance of a Savonius rotor. *Journal of Solar Energy Engineering* 116(3): 148-152.

[5] Nakajima M., Lio S. and Ikeda T., 2008. The performance of Savonius rotor for environmentally friendly hydraulic turbine. *Journal of Fluid Science and Technology* (3): 420-429.

[6] Altan D., Atılgan M. and Özdamar A., 2008. An experimental study on improvement of a Savonius rotor performance with curtaining. *Experimental Thermal and Fluids Science* (32): 1673-1678.

[7] Khan M.H., 1978. Model and prototype performance characteristics of Savonius rotor windmill. *Wind Energy* (2): 75-85.

[8] Faizal M., Raiffuddin A. and Young-Ho L., 2010. On utilizing the orbital motion in water waves to drive a Savonius rotor. *Renewable Energy* (35): 164-16.

[9] Khan M.J. 2009. Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review. *Applied Energy* (86): 1823-1835.

[10] Khan M.J., Iqbal M.T., Quaicoe J.E., 2008. River current energy conversion systems: Progress prospects and challenges. *Renewable and Sustainable Energy* (12): 2177-2193.

[11] Khalid S.S., Liang Z. and Shah N., 2012. Harnessing tidal energy using vertical axis tidal turbine. *Research Journal of Applied Sciences, Eng. and Technology* 5(1): 239-252.

[12] Pete B. and W. Martin. 2015. Characterizing the near-wake of a cross-flow turbine. *Journal of Turbulence*, DOI: 10.1080/14685248.2014.1001852

[13] Gebreslassie M.G., Tabor G.R. and Belmont M.R., 2013. Numerical simulation of new type of cross-flow tidal turbine using open foam. *Renewable Energy – Elsevier* (50): 994-1004

[14] Kailash, Golecha T.I., Eldho, and Prabhu S.V., 2012. Performance study of modified Savonius water turbine with two deflector plates. *International Journal of Rotating Machinery* Vol 2012, Article ID 679247, 12 pages.

[15] Wahyudi B., Soeparman S., Wahyudi S. and Denny W., 2013. A simulation study of flow and pressure distribution patterns in and around of tandem blade rotor of savonius (TBS) hydrokinetic turbine model. *Journal of Clean Energy Technologies* 1(4): 286-291.

[16] Wahyudi B., Soeparman S. and Hoeijmakers H.W.M., 2015 . Optimization design of diffuser with moving deflector for hydrokinetic cross flow turbine rotor. *Energy Procedia* (68): 244-253.

[17] Shamsun N., 2011. *Torque and drag characteristics of a six blade Savonius rotor*. Thesis. Mechanical Engineering, BUET Bangladesh.

[18] Chauvin A. and D. Benghrib. 1989. Drag and lift coefficient evolution of a Savonius rotor. *Journal Experiments in Fluid* (8): 118-120.

[19] Sarma N.K., Biswas A. and Misra R.D., 2014. Experimental and computational evaluation of Savonius hydrokinetic turbine for low velocity condition with comparison to Savonius wind turbine at the same input power. *Energy Conversion and Management* 83: 88-98.

[20] João Vicente A., da Silva Júnior G.A., and Petry A.P., 2012. Discussion on the verification of the overlap ratio influence on performance coefficients of a Savonius wind rotor using computational fluid dynamics. *Renewable Energy* 38(1): 141-149.

[21] Kanzumba K., 2014. Techno-economic analysis of off-grid hydrokinetic-based hybrid energy systems for onshore/remote area in South Africa. *Energy* 68: 947-957.

[22] Koko, S.P., 2014. Techno-economic analysis of an off-grid micro-hydrokinetic river system as a remote rural electrification option. Thesis Dissertation. Central University of Technology, Free State, Bloemfontein, South Africa.